Three-dimensional finite element modelling of composite slabs for high speed rails

Mlilo, Nhlanganiso; Kaewunruen, Sakdirat

DOI:
10.1088/1757-899X/280/1/012019

IOP Conference Series: Materials Science and Engineering

PAPER • OPEN ACCESS

Three-dimensional Finite Element Modelling of Composite Slabs for High Speed Rails

To cite this article: Nhlanganiso Mlilo and Sakdirat Kaewunruen 2017 *IOP Conf. Ser.: Mater. Sci. Eng.* **280** 012019

IOP Conf. Series: Materials Science and Engineering **280** (2017) 012019 doi:10.1088/1757-899X/280/1/012019

Three-dimensional Finite Element Modelling of Composite Slabs for High Speed Rails

Nhlanganiso Mlilo[1] and Sakdirat Kaewunruen[1,2]

[1] Department of Civil Engineering, The University of Birmingham, U.K.
[2] Birmingham Centre for Railway Research and Education, The University of Birmingham, U.K

nxm692@student.bham.ac.uk
KaewunrS@adf.bham.ac.uk

Abstract. Currently precast steel-concrete composite slabs are being considered on railway bridges as a viable alternative replacement for timber sleepers. However, due to their nature and the loading conditions, their behaviour is often complex. Present knowledge of the behaviour of precast steel-concrete composite slabs subjected to rail loading is limited. FEA is an important tool used to simulate real life behaviour and is widely accepted in many disciples of engineering as an alternative to experimental test methods, which are often costly and time consuming. This paper seeks to detail FEM of precast steel-concrete slabs subjected to standard in-service loading in high-speed rail with focus on the importance of accurately defining material properties, element type, mesh size, contacts, interactions and boundary conditions that will give results representative of real life behaviour. Initial finite element model show very good results, confirming the accuracy of the modelling procedure

1. Introduction

Composite steel-concrete beams due to their high stiffness, reduced deflections and high span/depth ratio are very popular in multistorey buildings [1]. In the say way, composite steel-concrete slabs connected to steel beams by means shear connectors usually shear studs represent an economic and efficient solution commonly used in building applications [2]. The steel sheeting acts as permanent formwork during construction and after hardening of the concrete acts as external tensile reinforcement. Considerable amount of research has been undertaken to date on the linear and nonlinear behaviour of composite floor beams using FE analysis [1]. Adoption of precast composite steel-concrete slab panels to resist rail loading was considered by Griffin [3] to address the problem of premature degradation associated with timber transoms on the Sydney Harbour Bridge. In this research a FE analysis software, ABAQUS was used to model and envisage the behaviour of theoretically determined composite steel-concrete panels when subjected to both standard in-service and derailment loading. However, the application of precast composite steel-concrete panels due to their nature and often complex loading conditions they are subjected to, the behaviour, interaction and response of the constituent components is often complex and requires a much thorough study. Present detailed knowledge of the behaviour of precast steel-concrete composite slabs subjected to rail loading is limited and there is therefore need to carry out thorough and detailed investigations in order to gain more understanding to inform future research.

In FE modelling, assigning reliable material properties, interactions and boundary conditions to the model that adequately represent the real situation or prototype is critical. A number of researchers have extensively and successfully used ABAQUS to analyse and investigate complex non-linear response of steel-concrete composite structures including Lam and Al-Lobody [4] who used ABAQUS to simulate the structural behaviour of headed shear studs in a steel-concrete composite beam and found that the FE results compared very well with experimental push-off test results while also accurately predicting the mode of failure of the headed shear studs. ABAQUS was also used to carry out an extensive parametric study to investigate the effect of changes in concrete strength and stud diameter on the capacity and behaviour of the shear connection [5]. In this research FE results were compared to those specified in design codes, namely; EC4 and AASTHO LRFD. A lot of other researchers have also developed FE models and successfully used ABAQUS.

2. Finite element modelling of composite steel-concrete slab

The accuracy of FE modelling greatly depends on the efficiency in simulating the non-linear behaviour of different materials within the composite structure when subjected to loading conditions.

2.1 Material Properties in ABAQUS

Steel-concrete composite slabs essentially consist of two structural materials i.e. concrete and steel which make up shear studs, profiled sheeting, reinforcing steel or prestressing steel and bridge stringer and it is important to carefully model the different components by accurately incorporating the stress-strain curves of these in the FE modelling. The accuracy of a FE analysis depends on constitutive laws used to define stress-strain characteristics of constituent materials [1].

2.1.1 Concrete The stress-strain curve of plain concrete in compression consists of two parts, the linear or elastic section which is defined by the modulus of elasticity, $E_{cm}$and the Poisson's ratio, v which is assumed to act up to a proportional limit of $0.4f_{cm}$. Beyond this point, the behavior is non-linear.

According to EC2 [6] the elastic deformation properties of plain concrete up to the characteristic compressive strength, f_{ck} of 90MPa can be accurately defined by the modulus of elasticity, E_{cm} in GPa, as:

$$E_{cm} = 22[(f_{cm})/10]^{0.3} \tag{1}$$

where
$$f_{cm} = (f_{ck} + 8)\ MPa \tag{2}$$

For this model plain concrete with a characteristic compressive cylinder strength, f_{ck} of 50MPa has been used and the modulus of elasticity has been calculated as 37GPa. The Poisson's ratio of concrete will be adopted as 0.2 for uncracked concrete and 0 for cracked concrete as detailed in EC2 [6].

There are several ways of modelling the plastic characteristics of concrete in ABAQUS including; plastic, cap plasticity, concrete damaged plasticity and concrete smeared cracking among many others. The non-linear compressive behaviour part of the stress-strain curve can be accurately found from the equation below [7].

$$\sigma_c = \frac{f_c' \gamma(\varepsilon_c/\varepsilon_c')}{\gamma - 1 + (\varepsilon_c/\varepsilon_c')} \tag{3}$$

where
$$\gamma = \left|\frac{f_c'}{32.4}\right|^3 + 1.55 \tag{4}$$

and
$$\varepsilon_c' = 0.002 \tag{5}$$

For concrete in tension, the stresses are assumed to increase linearly up to a point when the concrete cracks and after this they decrease linearly to zero [1].

2.1.2 Steel (Shear Studs, Profiled Steel Sheet, Reinforcing Steel and Bridge Stringer) The stress-strain behaviour of all the steel components i.e. shear studs, profiled sheet, bridge stringer, reinforcing steel is initially elastic after which yielding and strain hardening develops. Figure 1 below depicts the bi-linear stress-strain behaviour of steel sheeting and headed shear studs [3] while figure 2 depicts the expected strain-strain behaviour of bridge stringers and the reinforcing steel [1]. The stress-strain behaviour of all steel components is assumed to be the same in tension and compression.

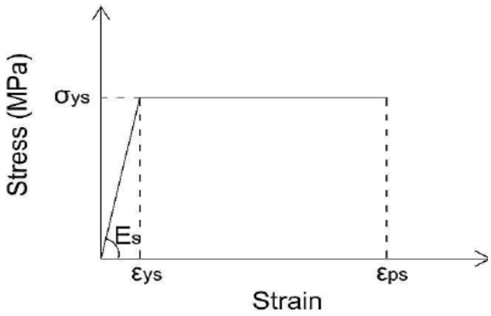

Figure 1. Bi-linear stress-strain curve

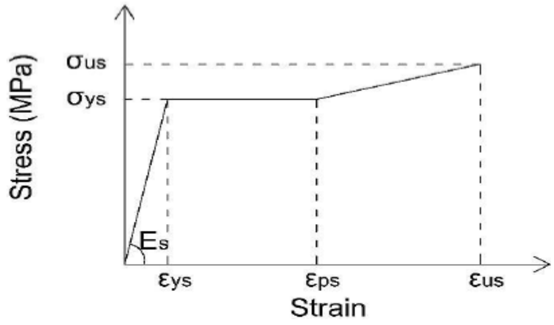

Figure 2. Tri-linear stress-strain curve

Table 1 below represents a summary of the component properties as well as the ratios for determining ultimate stress (σ_{us}), plastic strain (ε_{ps}) and ultimate strain (ε_{us}) [1].

Table 1. Summary of steel component properties

Component	*Young's Modulus E_s (GPa)*	*Poisson's Ratio, v*	*Yield Stress (f_y) MPa*	σ_{us} *(MPa)*	ε_{ps}	ε_{us}
Shear Studs	200	0.3	420	-	$25\varepsilon_{ys}$	-
Steel Sheeting	200	0.3	550	-	$20\varepsilon_{ys}$	-
Bridge Stringer	200	0.3	300	$1.28\varepsilon_{ys}$	$10\varepsilon_{ys}$	$30\varepsilon_{ys}$
Steel Reinforcement	200	0.3	500	$1.28\varepsilon_{ys}$	$9\varepsilon_{ys}$	$40\varepsilon_{ys}$

2.2 Finite Element Type and Mesh

In ABAQUS elements can be modelled as either two-dimensional or three-dimensional solid, shell, truss or beam elements. The stresses, strains as well as deformations are determined at the intersection of elements and because of this, to allow for an accurate analysis it is very important to select both the correct element type and element size. Solid, three-dimensional, eight node element with linear approximation of displacements, three translational degrees of freedom and reduced integration with hourglass control (C3D8R) can be used for linear and nonlinear models and is capable of handling large deformations, accommodate plasticity while accurately incorporating contact properties [8].

Mirza [1, 8] successfully used the C3D8R element to model the concrete slab and the steel beam and their use improved the rate of solution convergence.

To model thin walled sections, for example the profiled sheeting, it is recommended to use the S4R shell elements [1, 9 and 10]. These are four-node elements with 6 degrees of freedom per node and like the C3D8R incorporate reduced integration and when used by Mirza [1] to model the profiled steel sheeting provided accurate solutions and permitted quadratic deformation over the four nodal coordinates. The T3DR element which is a three-dimensional, two node truss element is used to model the steel reinforcement.

A good quality mesh is a very important in FE analysis as the shape and size of elements can significantly affect the results. A good balance needs to be achieved when the mesh size is being selected, the mesh needs to be fine enough to allow for good detail especially in the areas of most interest but at the same time not too fine as this can result in the analysis taking a lot of time to complete [1]. So in order to reduce the time for analysis, a fine mesh can be applied at regions where

IOP Conf. Series: Materials Science and Engineering **280** (2017) 012019 doi:10.1088/1757-899X/280/1/012019

there are interactions for example around the interface between the shear studs and concrete and around the interface between the shear studs and the steel beam/sheeting with a much coarser mesh applied everywhere else [5]. To ensure that a good model is produced which produces accurate results it is necessary to carry out a mesh sensitivity analysis. Table 2 below shows the adopted element types for each component as well as the optimum FE mesh sizes for the model which gave deflection results closest to the theoretically determined deflection.

Table 2. Summary of element type and mesh size

Component	*Element Type*	*Mesh Size*
Concrete	C3D8R	40
Shear Studs	C3D8R	40
Steel Sheeting	S4R	50
Bridge Stringer	C3D8R	40
Steel Reinforcement	T3DR	84.7

2.3 Contact and Interactions

Correct and accurate simulation of the interacting surfaces and parts within FE modelling is very important in order to obtain accurate results which reflect real life behaviour. The stability and adequacy of a FE model representing composite structures are influenced by the accurate definition of contact properties between the steel components and the concrete [2]. In structures like composite steel-concrete slabs where there are many interacting components, this is even more important. According to ABAQUS, there are many ways of defining interface interaction and the two most common are the constraint toolset and the interaction toolset. Surfaces are defined as master or slave with the stiffer material being assigned as a master and the other being assigned a slave. Reinforcing steel in concrete can sufficiently be modelled using the embedded technique. A summary of the surface designation and interface type is shown on the table 3 below.

Table 3. Surface and interface designation

Interacting Components	*Slave Surface*	*Master Surface*	*Interface Type*
Shear Stud - Concrete	Concrete	Shear Stud	Tie Constraint
Shear Stud – Steel Sheeting	Shear Stud	Steel Sheeting	Tie Constraint
Steel Sheeting – Stringer(weld)	Stringer	Steel Sheeting	Tie Constraint
Steel Sheeting - Stringer	Stringer	Steel Sheeting	Surface to Surface
Steel Reinforcement - Concrete	Concrete	Steel Reinforcement	Embedded
Concrete – Steel Sheeting	Concrete	Steel Sheeting	Surface to Surface

2.4 Boundary conditions and load application

Correct application of boundary conditions is crucial in FE modelling and the application should as much as possible accurately represent real-life situations otherwise, the model will not produce accurate results. In ABAQUS, boundary conditions can be applied to part edges, nodes or surfaces to define their translational or rotational response. If the models are symmetrical it is possible to either half or quarter them in order to reduce analysis time. When this is done the cut surfaces must have boundary conditions applied to them to represent the symmetry required. Several studies [1, 4, 5 and 10] investigating various aspects of composite slab behaviour produced accurate results when they used the above technique. The in-service model was considered symmetrical and is shown in figure 3.

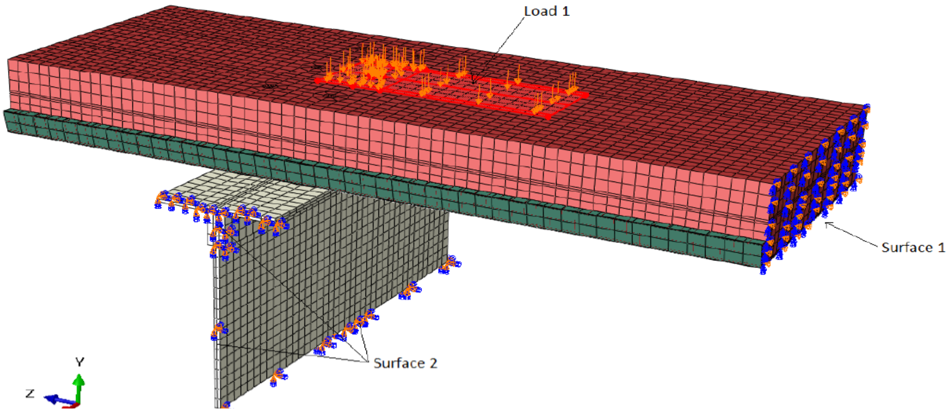

Figure 3. Load application and boundary conditions for the in-service half symmetrical model

Symmetric boundary conditions are applied to surface 1 and the nodes of the steel sheeting and concrete that lie on this surface are restricted from any translation in the z-direction. Surface 2 represents the cut edges of the bridge stringer and all the nodes on surface 2 are represented as encastre meaning that they are restrained from rotation and translation in all directions. The load combination which produced the worst loading case for bending moments was adopted for the simulation. The train load was applied as pressure load over the rail pads area and the factored dead load of the panel was also applied over the entire surface of the panel as the pressure load.

3. Results and conclusion

Figure 4 below represents the initial results from a symmetrical half in-service model developed following the above mentioned procedure.

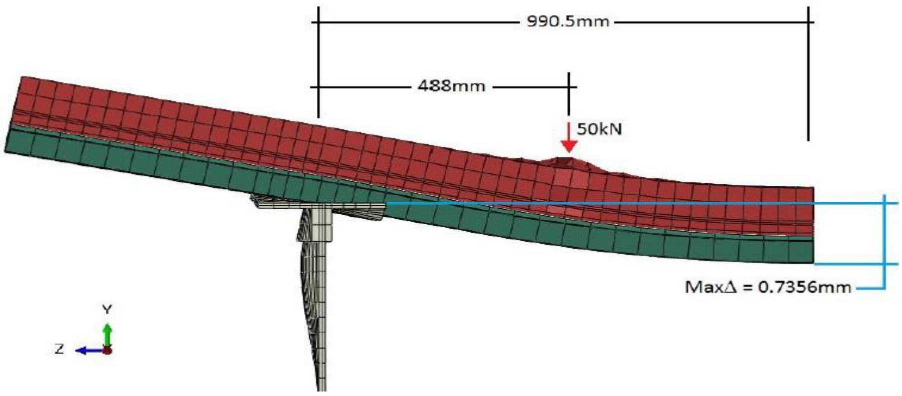

Figure 4. ABAQUS model showing in-service panel deflection

As shown on the figure above, the mid-span deflection which resulted from the ABAQUS model was 0.7356mm and comparing this to the theoretically determined deflection of 0.8137mm, it is clear that the results were comparable, i.e. 10.6% variation.

The maximum stresses developed in all the component parts compared to their corresponding yield stresses are summarised in the table 4 below.

Table 4. Limit strength summary of in-service panel

Component	*Material Yield Strength (MPa)*	*Maximum Stress developed (MPa)*	*Design ratio*
Concrete (in compression)	50	45	1.11
Concrete (in tension)	5.3	10.66	0.50*

Steel Sheeting	550	226	2.43
Stringer	300	133	2.26
Steel Reinforcement	500	32	15.63
Shear Studs	420	244	1.77

The design ratio is obtained by dividing the material yield strength by the corresponding maximum stress developed in the component. Looking at the design ratios, it is clear that all the components are adequate to resist the design loads apart from the concrete in tension which develops maximum tensile stress of 10.66MPa which is well over its maximum axial tensile strength, f_{ctk} of 5.3 MPa given in EC2 [6], implying that concrete does crack. However, upon cracking the steel sheeting continues to carry the tensile load. Figure 5 shows the concrete node (shown with arrow) which develops the maximum tensile stress.

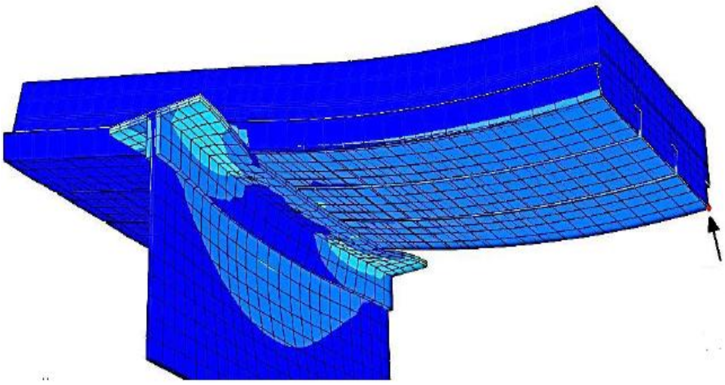

Figure 5. Critical node area in concrete

These results give us the confidence that the modelling procedure has been accurate and can be trusted as the theoretically designed in-service panel satisfies the ULS loading conditions.

References

[1] Mirza O 2008 *Behaviour and design of headed stud shear connectors in composite steel-concrete beams – PhD Thesis* (Sydney: University of Western Sydney)
[2] Tahmasebinia F, Ranzi G and Zona A 2012 *Journal of Constructional Steel Research* **80** 394-411
[3] Griffin D W P 2013 *Design of precast composite steel-concrete panels for track support: For use on the Sydney Harbour Bridge – BEng Thesis* (Sydney: University of Western Sydney)
[4] Lam D and El-Lobody 2001 *Structural Engineering, Mechanics and Computation* **1** 401-8
[5] Nguyen H T and Kim S E 2009 *Journal of Constructional Steel Research* **65** 1909-20
[6] British Standards Institute 2008 *BS EN 1992-1-1:2004 Eurocode 2: Design of concrete structures*
[7] Carreira D and Chu K 1985 *Journal of ACI Structures* **82** 797-804
[8] Mirza O and Uy B 2009 *Journal of Constructional Steel Research* **65** 662-74
[9] ABAQUS user's manual, version 6.14
[10] Shanmugam N E, Kumar G and Thevendran V 2002 *Finite Elements in Analysis and Design* **38** 579-99